

N82-11491

G3/39 08253

A circular stamp from the Massachusetts State Police. The outer ring contains the numbers 1 through 31, representing the days of the month. The center of the stamp contains the text: "RECEIVED", "MASS STATE FACILITY", and "ADDRESS DEPT.". The date "11/21/81" is stamped across the center.

NASA

INTEGRATED ANALYSIS OF ENGINE STRUCTURES

by

Christos C. Chamis
NASA Lewis Research Center
Cleveland, Ohio 44135

ABSTRACT

The need for light, durable, fuel efficient, cost effective aircraft requires the development of engine structures which are flexible, made from advanced materials (including composites), resist higher temperatures, maintain tighter clearances and have lower maintenance costs. The formal quantification of any or several of these requires integrated computer programs (multilevel and/or interdisciplinary analysis programs interconnected) for engine structural analysis/design. Several integrated analysis computer programs are under development at Lewis Research Center. These programs include: (1) COBSTRAN-Composite Blade Structural Analysis, (2) CODSTRAN-Composite Durability Structural Analysis, (3) CISTRAN-Composite Impact Structural Analysis, (4) STAEBL-Structural Tailoring of Engine Blades, and (5) ESMOSS-Engine Structures Modeling Software System. Three other related programs, developed under Lewis sponsorship, are described.

INTRODUCTION

NASA Lewis Research Center is now formulating a major Engine Structures Program with emphasis on advanced structural analysis, structural dynamics, structural aspects of aeroelasticity, and life prediction of turbine engine structures and structural components. These components can be made from conventional materials (used as bill of materials), advanced materials (utectics, directional solidified) and fiber composites. The general objective of this program is to develop the methodology to permit the rational structural design and analysis of components for advanced gas turbine engines as well as the overall engine system. Structural models will be developed to provide analysis capability which will assure the structural integrity of the part and to provide the designer with a method of optimizing component designs for maximum performance at minimum weight and cost. Engine system structural models will also be developed to analyze the behavior of the entire engine as a complex, interacting dynamic system. The models will incorporate overall displacements and distortions as well as component-to-component interactions due to steady-state and transient thermal and mechanical loads. The combined thermal-mechanical response of hot section components and other static or rotating engine structural components under various operating conditions such as takeoff, cruise, maneuver, and landing need be determined. Fracture mechanics methods specifically related to engines, such as elevated temperature, nonlinear crack growth, and failure prediction methods, will also be developed. Fundamental and common to all these developments is the precise geometric and analytical model description of engine structures at several assembly levels from individual parts, through the component, the substructure and the entire engine.

Both the precise geometric modeling and the analytical models at the several different levels of the engine structure require integrated computer programs. Integrated computer programs consist of independent programs each with its own input/output FORMATS communicating with each other through a common program called the executive module. This paper describes, briefly, integrated analysis, optimization, and modeling computer programs under development at Lewis Research Center. The length of the discussion on each of these integrated computer programs reflects the status of the development at both the integration and module levels. Most of these integrated programs are mainly to be used as in-house research tools and are not amenable to dissemination in general.

OPEN-ENDED MODULAR PROGRAMS INTEGRATION

Integrated engine structural analysis encompasses the upward integration of several interdisciplinary methodologies. This is illustrated schematically in Fig. 1 assuming that each methodology is in the form of a computer program module. For example, the composite mechanics module includes composite micromechanics, composite macromechanics, combined stress failure criteria and laminate theory in general. A specific integrated capability will be described later. Also, advanced structural analysis may include continuum mechanics methods, approximate methods, singularity finite elements, interactive finite elements etc. The significant points to be noted in Fig. 1 are: (1) The apex of the upward integration is the validation of structural concepts. (2) The structure is open-ended at all levels. (3) Integration takes place laterally as well as vertically. (4) Each module integrates more fundamental methodologies. (5) The integrated programs to be described herein lead directly to the apex.

In general, it is not necessary to have as comprehensive an integrated analysis as shown in Fig. 1 in order to validate a structural design concept of a simple component. The upward integrated structure shown in Fig. 1 does not preclude the validation process of a structural design concept for a simple component. In this case the progressively higher level integration modules can be imagined to be either blank or bypass juncture points.

COBSTRAN: COMPOSITE BLADE STRUCTURAL ANALYSIS

COBSTRAN has been under development at Lewis for about eight years. The documentation and software for a completed version are about ready for public release through COSMIC (Computer Software Management and Information Center, University of Georgia, Athens, Georgia).

COBSTRAN is an analysis procedure streamlined for composite blade analysis. The blades can be solid, hollow, solid with cavities and combinations of these. The analysis procedure consists of using NASTRAN in conjunction with composite mechanics coupled within pre- and postprocessors. The pre- and postprocessors are especially designed to automate the large amount of information needed to analyze fiber composite compressor blades via

NASTRAN. The preprocessors are used to generate three types of information required as input for NASTRAN. Briefly, these types are:

1. Finite element representation, nodal coordinates, nodal thickness, and boundary conditions.
2. Nodal pressures and temperatures.
3. Anisotropic material properties generated by the composite mechanics module using input constituent properties, fiber volume ratio, void ratio, ply orientation and ply contours.

The postprocessor is used to reduce NASTRAN output information to input for the composite mechanics module to calculate ply stresses and/or strains and the corresponding margins of safety. The NASTRAN output information, in general, consists of nodal displacements, element force resultants, element stresses and the corresponding principal stresses, and frequencies for various vibration modes. The overall blade untwist and uncamber are determined from nodal displacements at the tip.

For the analysis of a proposed blade, triangular or quadrilateral finite element representations can be used. The elements includes bending and membrane responses, centrifugal forces, and anisotropic material properties. These elements are identified as CTRIA2 and CQUAD2 in the NASTRAN library of elements. A schematic of the flow chart of COBSTRAN is shown in Fig. 2. A COBSTRAN finite element model for a composite windmill blade is shown in Fig. 3. Selected results from this model are shown in Fig. 4. Additional analyses are described in references 1, 2, and 3.

CODSTRAN: COMPOSITE DURABILITY STRUCTURAL ANALYSIS

CODSTRAN has been under development at Lewis for about four years. CODSTRAN is an integrated computer program designed to predict defect growth and fracture of composite structural components subjected to engine service load and environmental conditions. CODSTRAN is modular and open-ended. Present modules may be replaced as improved ones become available and new ones can be added as they are developed.

The organization of CODSTRAN is shown schematically in the flow chart, Fig. 5. As can be seen in this figure CODSTRAN consists of seven major modules: Executive, Input, Output, Analysis, Composite Mechanics, Fracture Criteria, and Life Prediction. Communication between modules is through the Executive module.

The functional features of the various modules will eventually provide CODSTRAN with the following capabilities:

1. Durability assessment of large structures and complex structural parts from composites.
2. Durability of components with regular and irregular defect geometry.
3. Accurate predictions of stress states near defects and discontinuities.

4. Structural response due to static, cyclic, transient impact and thermal loads.
5. Evaluation of defects in all types of fibrous composites and hybrids.
6. Assessment of environmental (temperature and moisture) effects on fibrous composites with defects.
7. Assessment of the effects of geometric and material nonlinearities on defect growth and fracture.
8. Criteria for static, cyclic and dynamic fracture.
9. Ply, interply and free-edge interlaminar failure criteria.
10. Prediction of laminate level fracture when defects are present.
11. Prediction of residual stress effects on defect growth.
12. Evaluation of in situ ply strength effects on defect growth.

It is believed at this time that these capabilities are essential for reliably predicting the structural integrity of fibrous composite components and for assessing the associated durability.

CODSTRAN pilot model results with environment effects are shown in Fig. 6. CODSTRAN predicted results are compared with experimental data in Fig. 7. These two figures illustrate the usefulness and potential of CODSTRAN like analysis capability for composites. CODSTRAN is described in more detail in reference 4.

CISTRAN: COMPOSITE IMPACT STRUCTURAL ANALYSIS

CISTRAN has been under development at LeRC since the fall of 1979. It is the least developed of the integrated computer programs for composite structural analysis. The flow chart schematic of CISTRAN is shown in Fig. 8. The various analysis modules in CISTRAN are divided into two groups. One group deals with the local impact and includes wave mechanics and a special three dimensional finite element. The analysis methodology and computer programs were initiated in the early 1970's. The global analysis includes NASTRAN and a streamlined approximate modal analysis for blades. The corresponding methodology and experimental data required for dynamic analysis and dynamic fracture criteria have been generated since the mid 1970's. Additional discussion with typical results follows.

The wave mechanics methodology was developed under grant over a six year period (ref. 5). It uses classical wave mechanics to solve the impact depicted in Fig. 9 for pressure, flexural and shear waves. Typical results obtained are shown in Fig. 10. In addition to the impact depicted in Fig. 9, the methodology and requisite computer programs for oblique and edge impact have been developed and will be modules in CISTRAN.

The special finite element for local impact consists of a three-dimensional finite element computer code which was developed under contract (refs. 6 and 7) for determining the local response of components and structures which are subjected to high velocity impact. This code is based on the coupled Eulerian-Lagrangian formulations (mode) and has been given the acronym CELFE (Coupled Eulerian Lagrangian Finite Element). CELFE is structured to handle high velocity impact structural response in the presence of geometric and material nonlinearities, material flow, and anisotropic material behavior. CELFE has been coupled with NASTRAN in order to: (1) take advantage of already available capabilities in NASTRAN to solve large systems of equations which require out-of-core storage, and (2) to determine the far-field structural response of large structures subjected to local high velocity impact. Both the projectile and the target can be modeled using finite elements as depicted schematically in Fig. 11.

The approximate modal analysis for global impact response consists of a computerized capability which was also developed under contract (ref. 8). This capability is identified as "Interactive Multi-Mode Blade Impact Analysis" and can be used to determine the structural dynamics response of turbine engine fan blades subjected to soft-body (bird) impacts. Both the theoretical development and resulting computer program of this capability are an outgrowth of two programs that were previously developed for the purpose of studying problems of a similar nature (a 3-mode beam impact analysis and a multi-mode beam impact analysis). The program utilizes a missile model that is interactively coupled with blade motion which is consistent with actual observations. The analysis capability in this computer program represents a significant improvement in the development of the methodology for evaluating potential fan blade materials and designs with regard to foreign object impact resistance. Compared to the 3-D finite element code, this program is limited to a linear response and uses superposition of mode shapes. It, therefore, uses considerably less computer time and is suitable for determining the overall (global) response of the blade, for preliminary designs, and for trade-off studies. Typical results from the approximate modal analysis are shown in Fig. 12. The direct time integration capability of NASTRAN is also used to determine the global response (ref. 9). Results that were obtained using this approach for a hybrid composite blade (Fig. 13) are shown in Fig. 14.

STAEBL: STRUCTURAL TAILORING OF ENGINE BLADES

STAEBL is being presently developed under contract (NAS3-22525) for the optimum design of engine fan blades. Advanced engine design require these blades to be lightweight, low cost and must endure extended airline service. STAEBL is a formalized optimum design procedure of engine blades to meet all the aerothermomechanical design requirements in an aircraft engine environment. Formalized optimum design procedures have been developed and used with considerable success for optimum structural design of linear static structures, and are now being developed and used with some success for the aeroelastic tailoring of fixed aircraft wings. In both of these general classes of problems: (1) The analyses deal mainly with static or quasi-static conditions; (2) the loading conditions are usually well defined; and (3) the required analyses have been well developed. On the other hand: (1) engine blades operate in dynamic conditions; (2) loading condition for

the blades are not well defined; (3) structural and structural dynamic analyses of engine blades are not well developed; (4) use of finite elements results in analysis is too time consuming to be used effectively in an optimum design procedure; and (5) engine blades are made from homogeneous materials which are not readily amenable to structural tailoring in general. As a result, analysts and/or designers have hesitated to develop optimization procedures for structural tailoring of engine blades even though they are keenly aware of the potential benefits derivable from such procedures.

STAEBL is designed to take advantage of available optimization procedures, approximate analyses to determine the aerothermomechanical response, the use of advanced design concepts, and the use of composite materials. The modular and integrated structure of STAEBL is shown in Fig. 15. The approximate analyses box includes analysis modules for: (1) vibration, (2) flutter, (3) stress, (4) foreign object ingestion, and (5) fatigue. No results are available as of this writing.

ESMOSS: ENGINE STRUCTURES MODELING SOFTWARE SYSTEM

The ESMOSS computer program is presently being developed under contract (NAS3-22767). It is a specialized software system that will construct geometric descriptive and discrete analytical models of engine parts, components and substructures which can be transferred to analysis programs (such as NASTRAN), or to displays (such as X-Y plotters) or graphics displays terminals. The program will also have the capability to locally alter or refine the geometry or generated mesh when analysis programs identify regions where improved accuracy is needed (such as near stress concentrations).

The ESMOSS software architecture is designed and developed in modular form with a central executive module through which the user controls all communication between and interaction of these modules (Fig.16). The modules are open-ended to allow for future expansion. The modules consist of: (1) a library of geometric shape generators, specially selected on the basis of their utility in analyzing engine structures; (2) a library of discretizing procedures to produce discrete analytical models from the geometric descriptions, including procedures for local mesh refinement (zooming and resizing); (3) an interfacing module to be used for joining the geometric descriptions and discretized models of structural parts into a structure; (4) component modules to create a component from component parts such as a fan blade from a separately generated airfoil, shroud, platform and root; and (5) substructure modules to assemble components into a complete substructure such as a fan stage from a disk and fan blades. The system will have separate modules to handle input and output data to and from stored files, external analysis programs and devices such as terminals and graphics display systems. Furthermore, the system will provide for a library of discretization criteria and optimization procedures for aiding the analyst constructing models. These may be used by the discretizing module during initial mesh generation, by the interfacing module to insure proper matching of the geometry and discretization at an interface, or to control mesh refinement during zooming and resizing. The capability of ESMOSS initially will consist of sufficient software to describe the geometric and NASTRAN finite element models of the following two engine substructures: (1) a complete fan stage and (2) a section of a burner liner. The fan stage consists of blades and blade disk

assemblies shown in Fig. 17 and 18. The burner liner is an axisymmetric complex shell a section of which is shown schematically in Fig. 19.

INTEGRATED PROGRAM FOR COMPOSITE HYDROTHERMOMECHANICAL (HDTM). RESPONSE

HDTM is a different level of an integrated program than those described previously. HDTM was specifically developed to predict the structural response of fiber composite structural components when these components are subjected to HDTM environments. Unique features of this integrated program are: (1) the integration starts from constituent properties, (2) the HDTM environment effects on thermomechanical behavior are incorporated at the constituents levels; and (3) the program is upward integrated using the theoretical disciplines: composite micromechanics, composite macromechanics, laminate theory, and structural analysis. The general structure of HDTM is depicted schematically in Fig. 20. Each discipline module requires appropriate input which is integrated locally to provide input information for the next module. The next module usually requires input not only from the lower level module, but also from other sources. This program has been used with considerable success at Lewis over the last four years (refs. 10 and 11).

OTHER INTEGRATED PROGRAMS

Three other major integrated computer programs for engine structural analysis have been developed under Lewis sponsorship: (1) Turbine Engine Transient Analysis (TETRA), (2) Aeroelastic and Dynamic Finite Element Analysis of a Bladed Shrouded Disk (ABASD) and (3) Nonlinear Transient Analysis for Fragment Containment (NONTRAN). TETRA uses the component element method to describe the transient structural response of the engine during and subsequent to blade-loss impact. TETRA was developed under a Lewis contract to General Electric Company, Aircraft Engine Business Group, ref. 12. ABASD couples internal flow aerodynamics (cascade theory) with the cyclic symmetry capability in NASTRAN. The cyclic capability is modified to account for centrifugal force stiffening. The coupling is packaged into a RIGID FORMAT for normal use with NASTRAN. ABASD was developed under a Lewis contract by Bell Aerospace Corporation, ref. 13. NONTRAN is a finite element based analysis accounting for both geometry (large deformation) and material viscoelastic behavior, nonlinearities. It traces the transient response from the time the fragment breaks off until it is contained. NONTRAN can handle plate-like structures with a variety of boundary conditions and with or without stiffeners. NONTRAN was developed under a Lewis grant at the Massachusetts Institute of Technology, ref. 14.

SUMMARY

The complexities of advanced engine structures require integrated computer analysis/design capabilities in the form of computer programs. Several integrated computer programs under various stages of development, at Lewis Research Center, are described. These include: (1) COBSTRAN-Composite Blade Structural Analysis; (2) CODSTRAN-Composite Durability Structural Analysis; (3) CISTRAN-Composite Impact Structural Analysis; (4) STAEBL-Structural Tailoring of Engine Blades; (5) ESMOSS-Engine Structures Modeling Software System; and (6) HDTM-Hydrothermomechanical response of composites. Three other integrated computer programs, developed under Lewis sponsorship, are also described.

ACKNOWLEDGMENTS

The development of integrated computer programs requires the "integrated" efforts of several individuals frequently with knowledge in different disciplines. I like to thank all my colleagues at Lewis and elsewhere who have contributed and continue to contribute to the development of our integrated programs for engine structural analysis. Also, this type of development has the tendency for dragging on and often taxing the patience of the supervisors. I like to express my sincere appreciation to all my supervisors for their understanding and patience.

REFERENCES

1. Chamis, C. C. and Lynch, J. E., "High-Tip Speed Fiber Composite Compressor Blades: Vibration and Strength Analysis," NASA TM X-71589, 1974.
2. Chamis, C. C. and Minich, M. D., "Structural Response of a Fiber Composite Compressor Fan Blade Airfoil," NASA TM X-71623, 1975.
3. Chamis, C. C., "Vibration Characteristics of Composite Fan Blades and Comparisons with Measured Data," NASA TM X-71893, 1976.
4. Chamis, C. C. and Smith, G. T., "CODSTRAN: Composite Durability Structural Analysis," NASA TM-79070, 1978.
5. Kim, B. S. and Moon, F. C., "Impact on Multilayered Composite Plates," Final Report, Cornell Univ., Ithaca, NY, April 1977. (NASA CR-135247)
6. Lee, C. H., "CELFE: Coupled Eulerian-Lagrangian Finite Element Program for High Velocity Impact, Part I- Theory and Formulation," LMSC-HREC-TR-D497204-PT-2, Lockheed Missiles and Space Co., Huntsville, AL, January 1978. (NASA CR-159462)
7. Lee, C. H., "CELFE: Coupled Eulerian-Lagrangian Finite Element Program for High Velocity Impact, Part II - Program User's Manual," LMSC-HREC-TR-D497204-PT-1, Lockheed Missiles and Space Co., Huntsville, AL, January 1978. (NASA CR-159396)
8. Alexander, A. and Cornell, R. W., "Interactive Multi-Mode Blade Impact Analysis," Final Report, Hamilton Standard, Windsor Locks, CT, August 1978. (NASA CR-159462)
9. Chamis, C. C. and Sinclair, J. H., "Analysis of High Velocity Impact on Hybrid Composite Fan Blades," NASA TM-79133, 1979.
10. Chamis, C. C., Lark, R. F., and Sinclair, J. H., "An Integrated Theory for Predicting the Hydrothermomechanical Response of Advanced Composite Structural Components," NASA TM-73812, 1977.
11. Chamis, C. C., Sinclair, J. H., and Lark, R. F., "Effects of Moisture Profiles and Laminate Configuration on the Hygrostresses in Advanced Composites," NASA TM-78978, 1978.

12. Callardo, V. C. et.al., "Blade Loss Transient Dynamic Analysis, Vols. I, II and III," R81AEG381, General Electric Company, Cincinnati, OH, June 1981. (NASA CR-165373)
13. Smith, G. C. C. and Elchuri, V., "Aeroelastic and Dynamic Finite Element Analyses of a Bladed Shrouded Disk," D2536-941001, Textron Bell Aerospace Company, Buffalo, NY, March 1980. (NASA CR-159728)
14. Spilker, R. A. et.al., "Finite-Element Nonlinear Transient Response Computer Programs PLATE 1 and CIVM-Plate 1 for the Analysis of Panels Subjected to Impulse or Impact Loads," ASRL-TR-154-14, Massachusetts Institute of Technology, Cambridge, MA, September 1980. (NASA CR-165215).

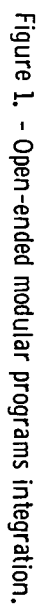


Figure 1. - Open-ended modular programs integration.

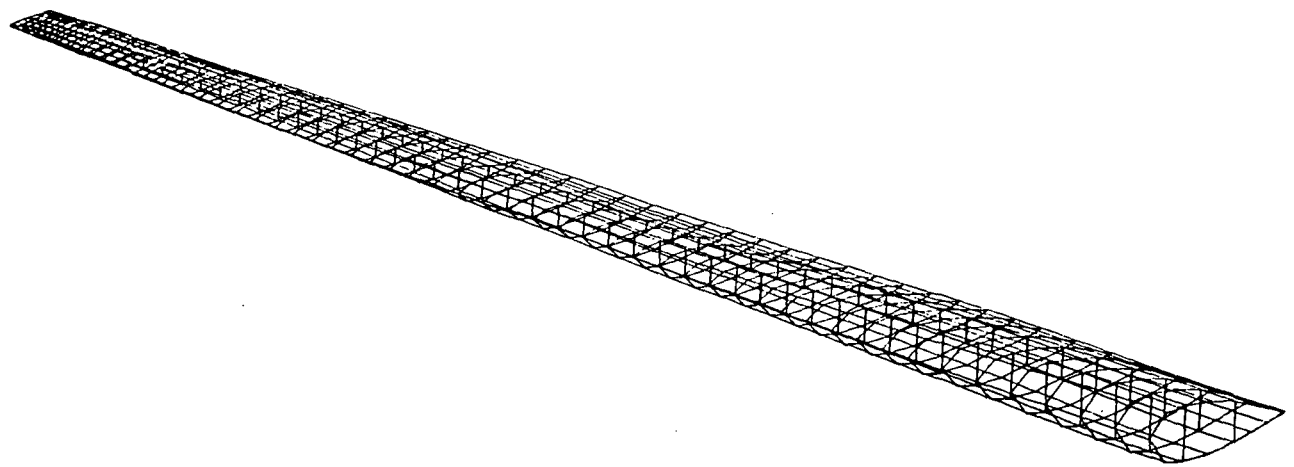
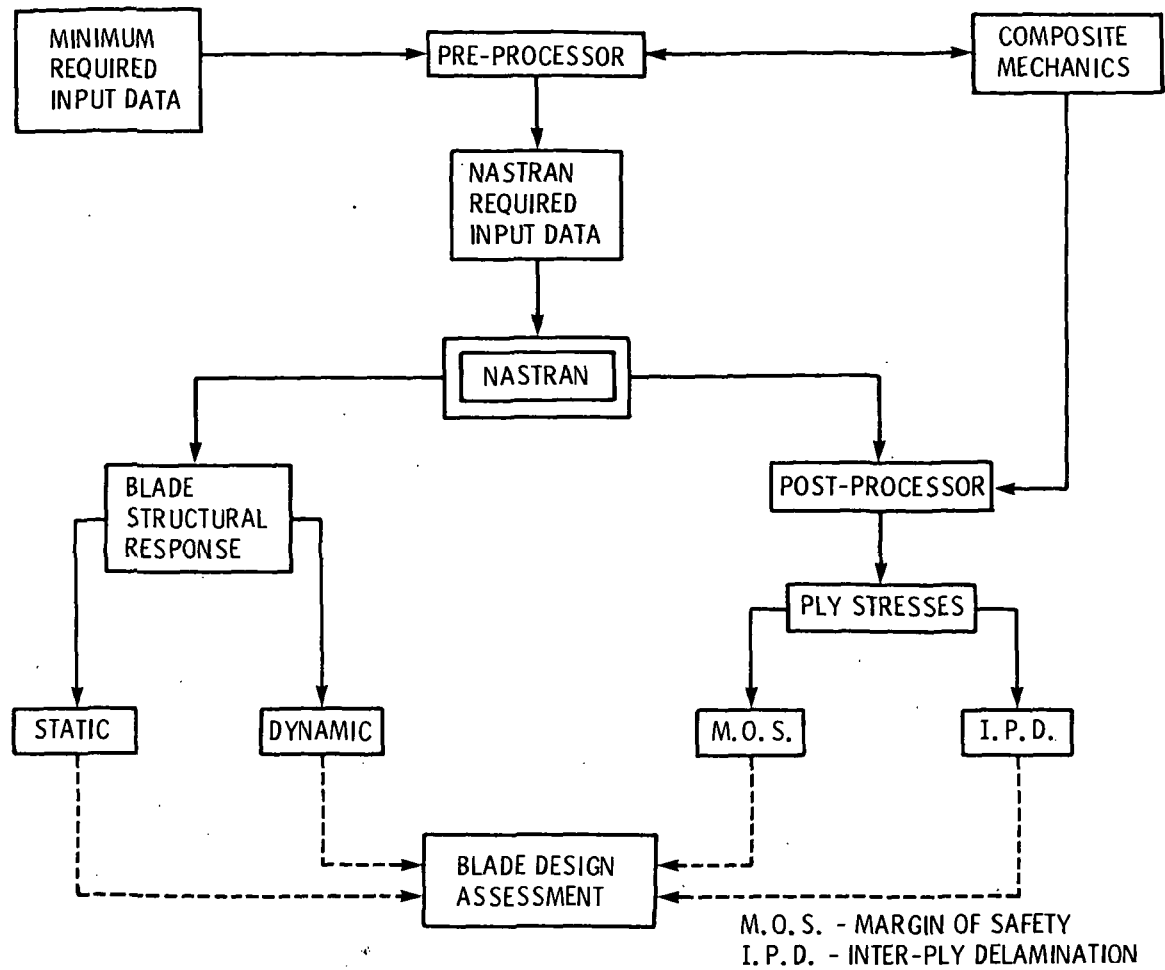


Figure 3. - Windmill composite blade COBSTRAN finite element model.

CRITICAL DESIGN LOAD CONDITION:

WIND SPEED 60 mph

ROTOR SPEED 45 rpm

BLADE DESIGN CONFIGURATION:

DOUBLE D-SPAR/SKIN

F. E. REPRESENTATION STATISTICS

NODES 290

DOF 1460

ELEMENTS 336

RESULTS:

MAX. STRESS < 20 ksi

WEIGHT/BLADE < 1000 lb

FUND. FREQ. \approx 112 cpm

BLADE TIP DISP. \approx 9 in.

Figure 4. - Wind mill composite blades. (Demonstrate preliminary design of fixed root E-glass/resin composite blade.)

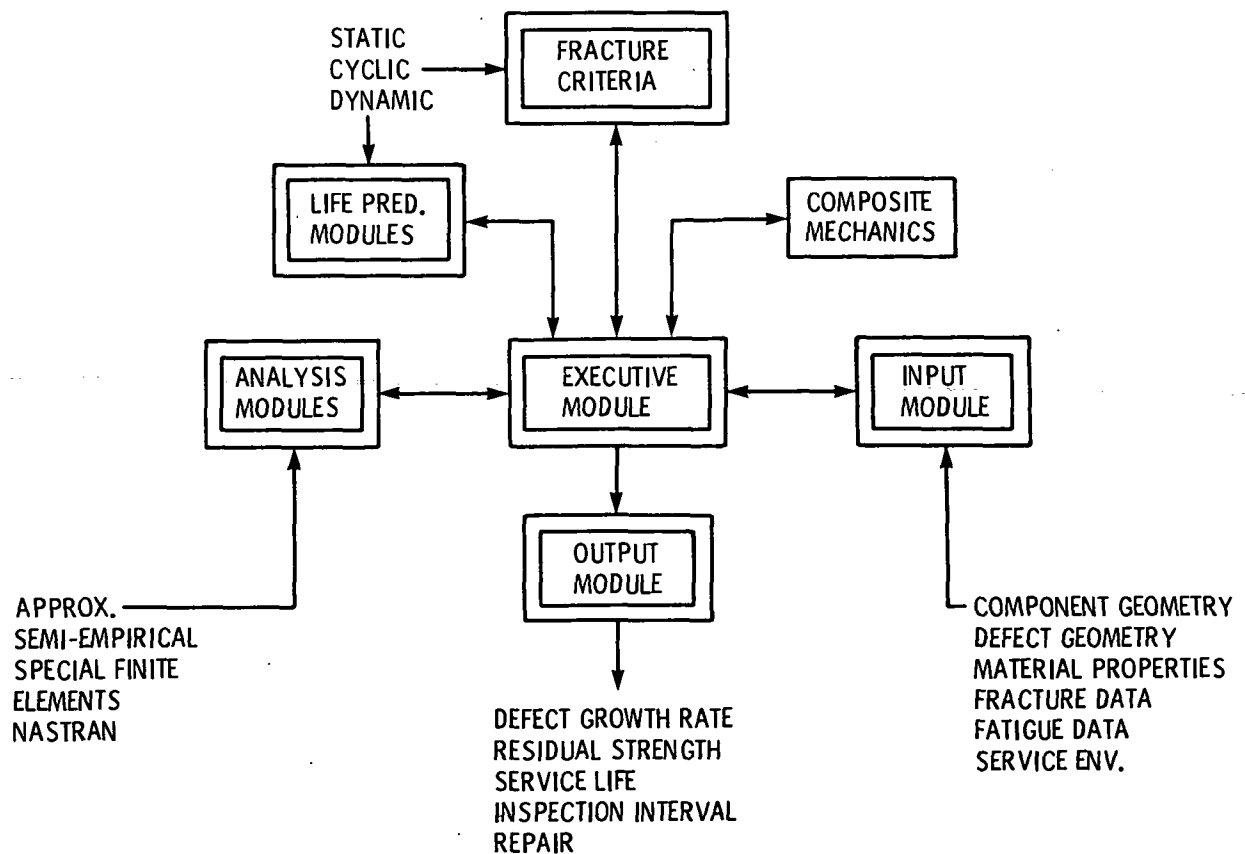


Figure 5. - CODSTRAN (Composite durability structural analysis.) (Conceptual program organization.)

COMPUTER PROGRAM
DESIGNED TO SIMULATE
DEFECT GROWTH AND
FRACTURE IN COMPOSITES
INCLUDING HYGROTHERMAL
EFFECTS

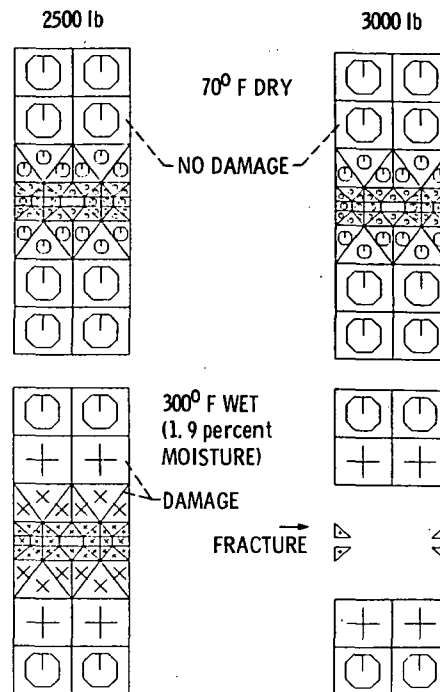


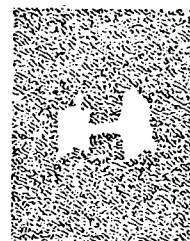
Figure 6. - CODSTRAN pilot model results with environmental effects.



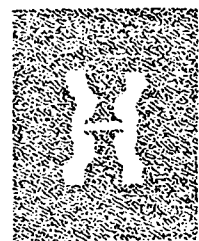
(a) NO LOAD



(b) LOAD EQUAL APPROXIMATELY
ONE-HALF FRACTURE LOAD.



C-SCAN RECORD



CODSTRAN

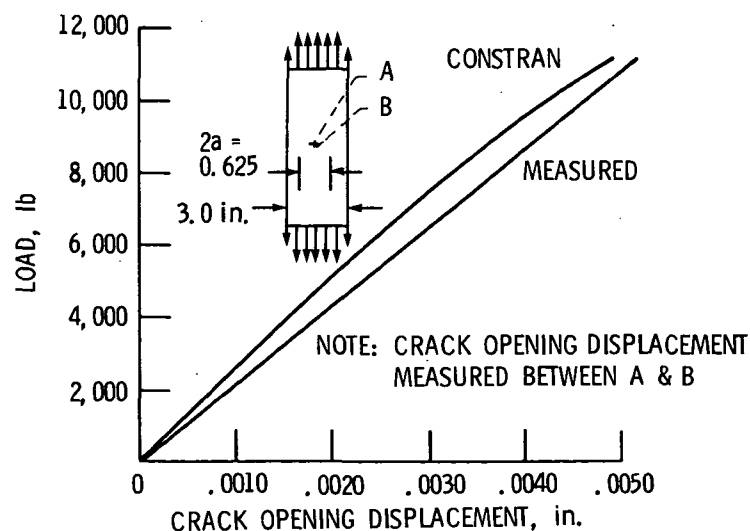


Figure 7. - CODSTRAN predicted results compared with exp. data.

ORIGINAL PAGE IS
OF POOR QUALITY

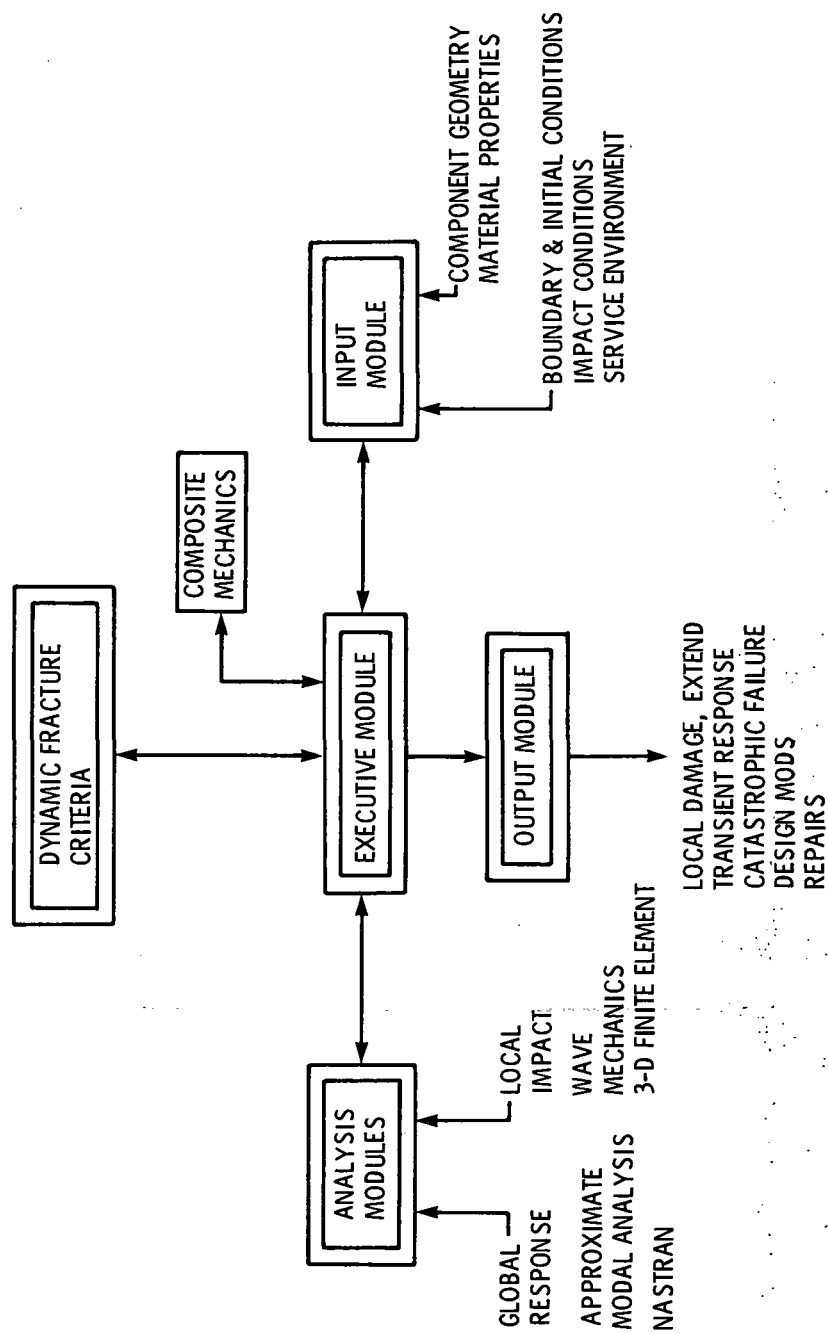


Figure 8. - CISTRAN (Composite impact structural analysis.) (Conceptual program organization.)

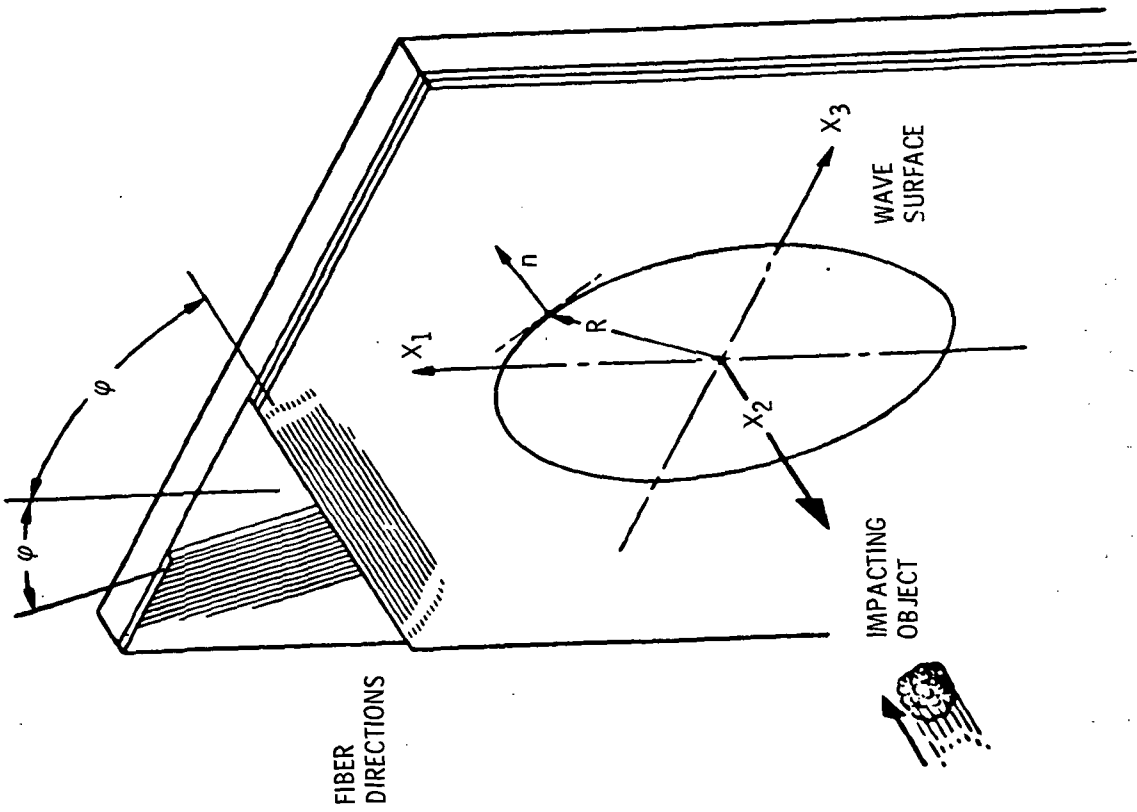


Figure 9. - Normal impact of fiber composite panels.

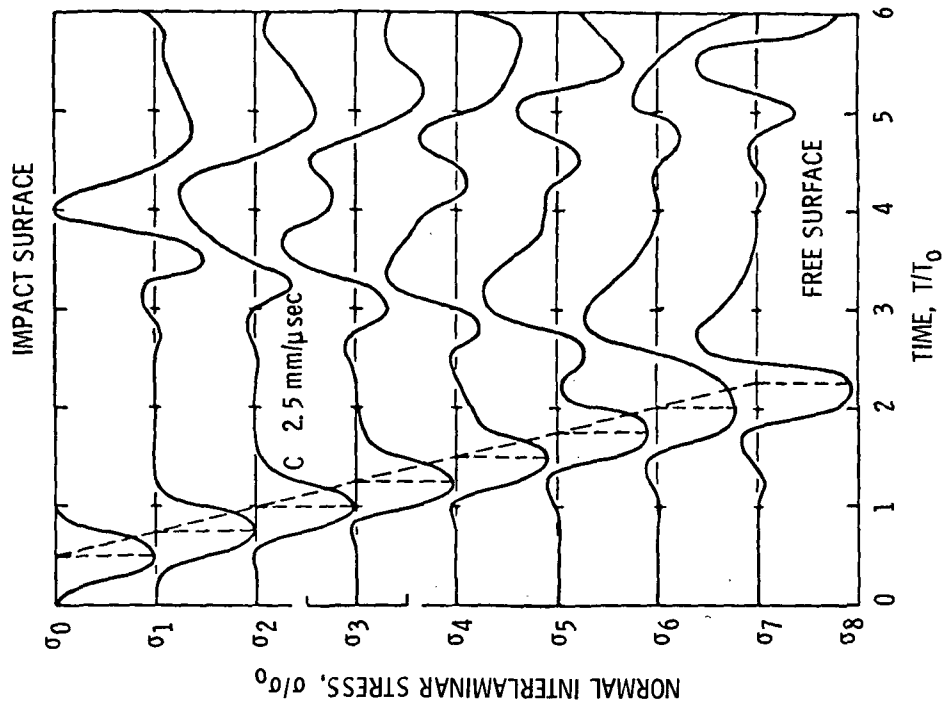


Figure 10. - Wave mechanics results for stress wave propagation in fiber composite laminates. (σ - interply layer between plies 1 and 2, etc.)

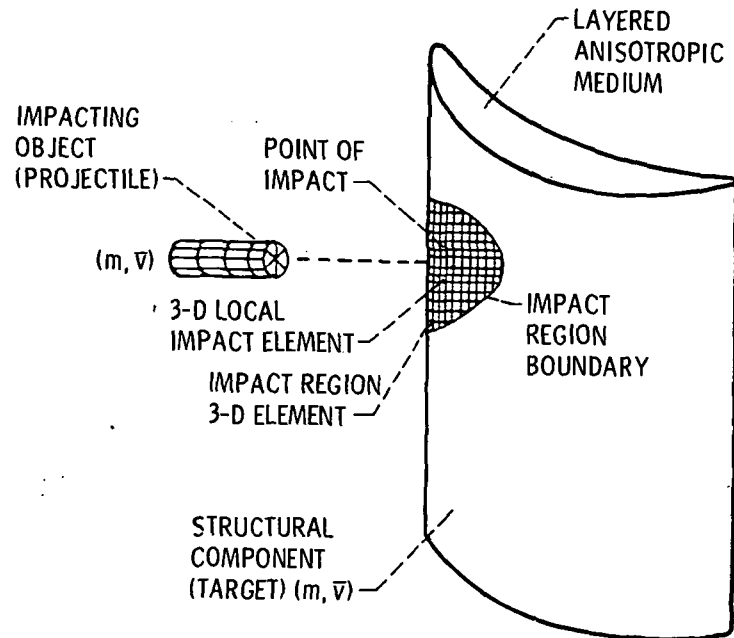


Figure 11. - 3-D High velocity impact via coupled eulerian lagrangian finite element (CELFE).

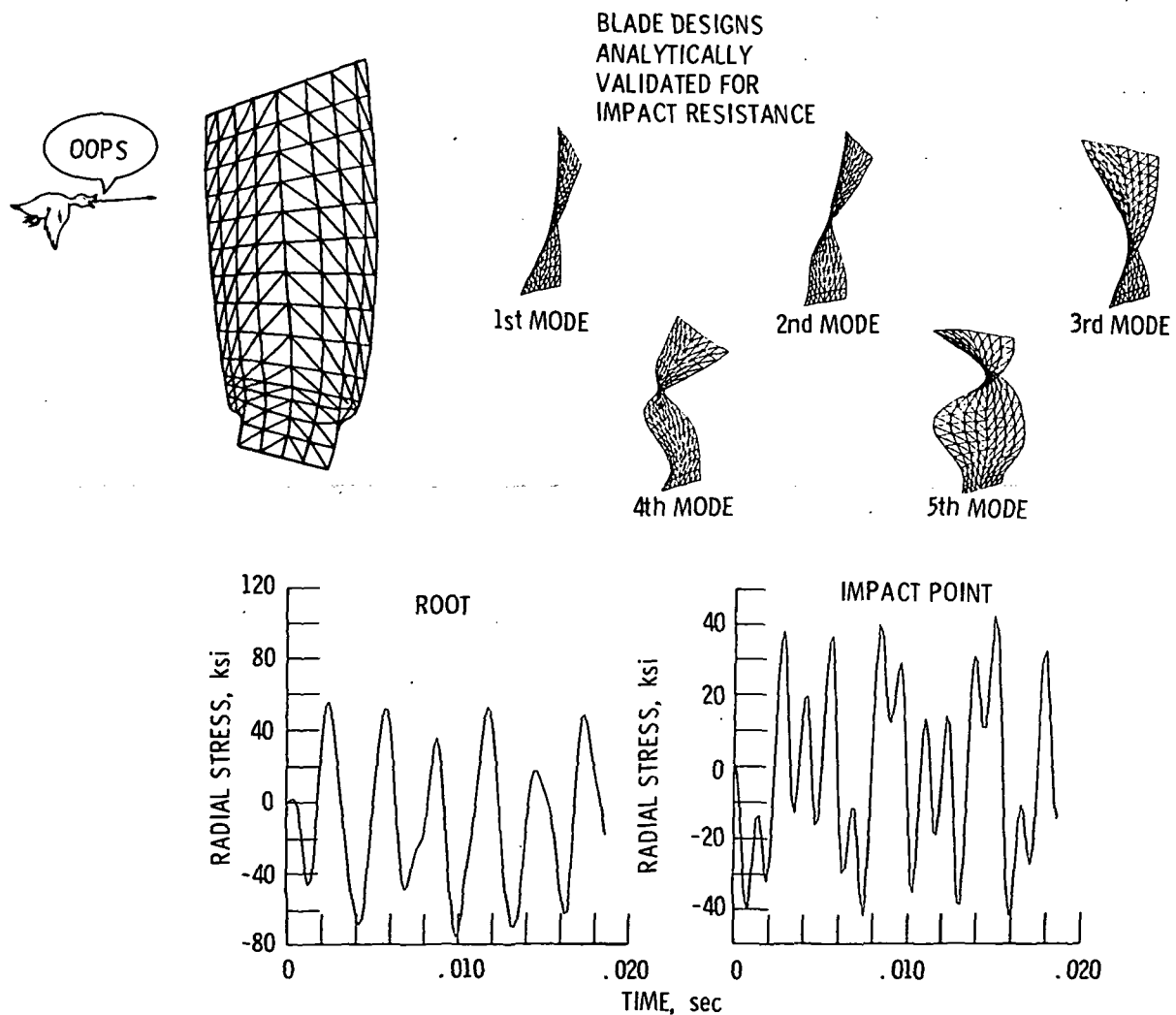


Figure 12. - Multi-mode blade impact analysis.

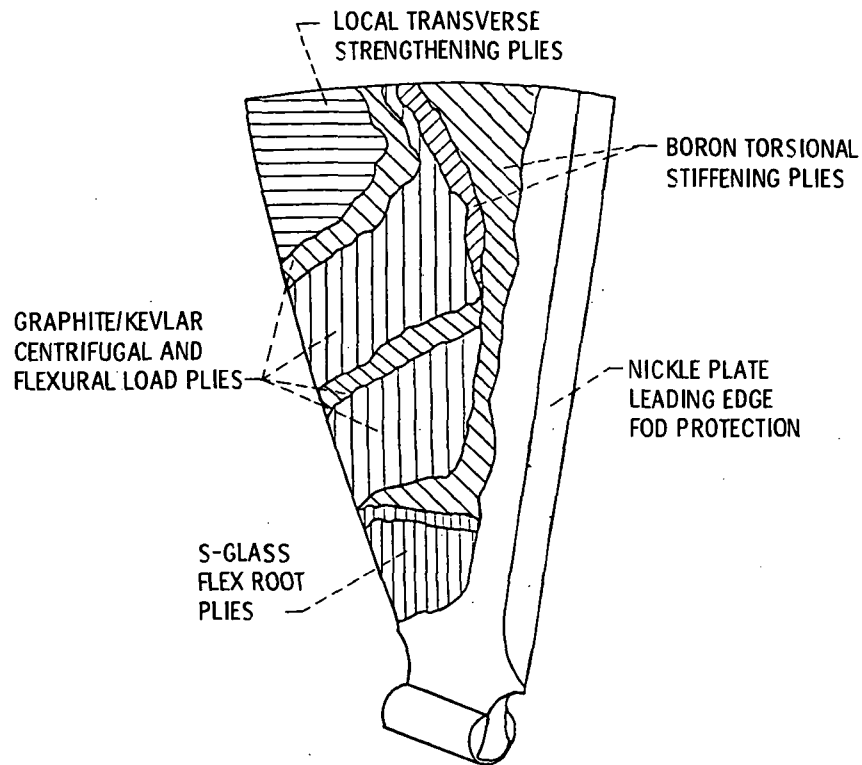


Figure 13. - Hybrid composite blade subjected to impact.

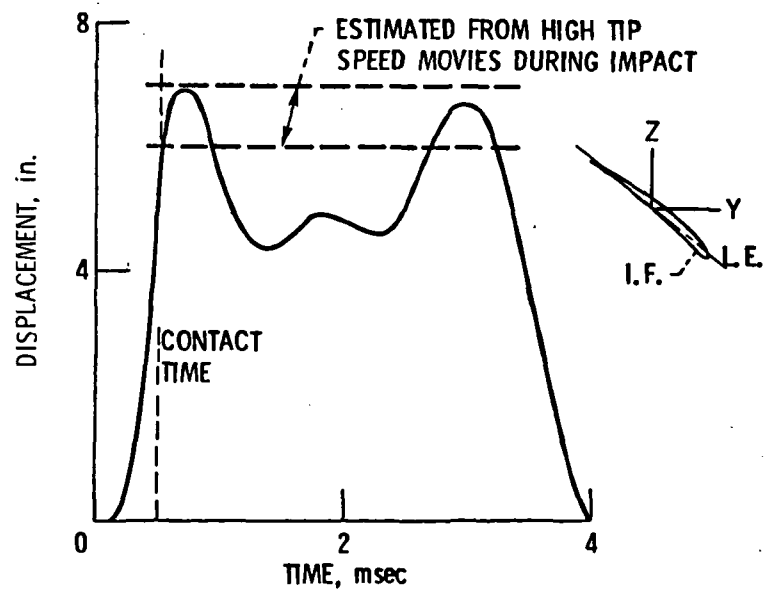


Figure 14. - Z-component of L.E. tip displacement. Hybrid composite fan blade, 2 lb bird impact.

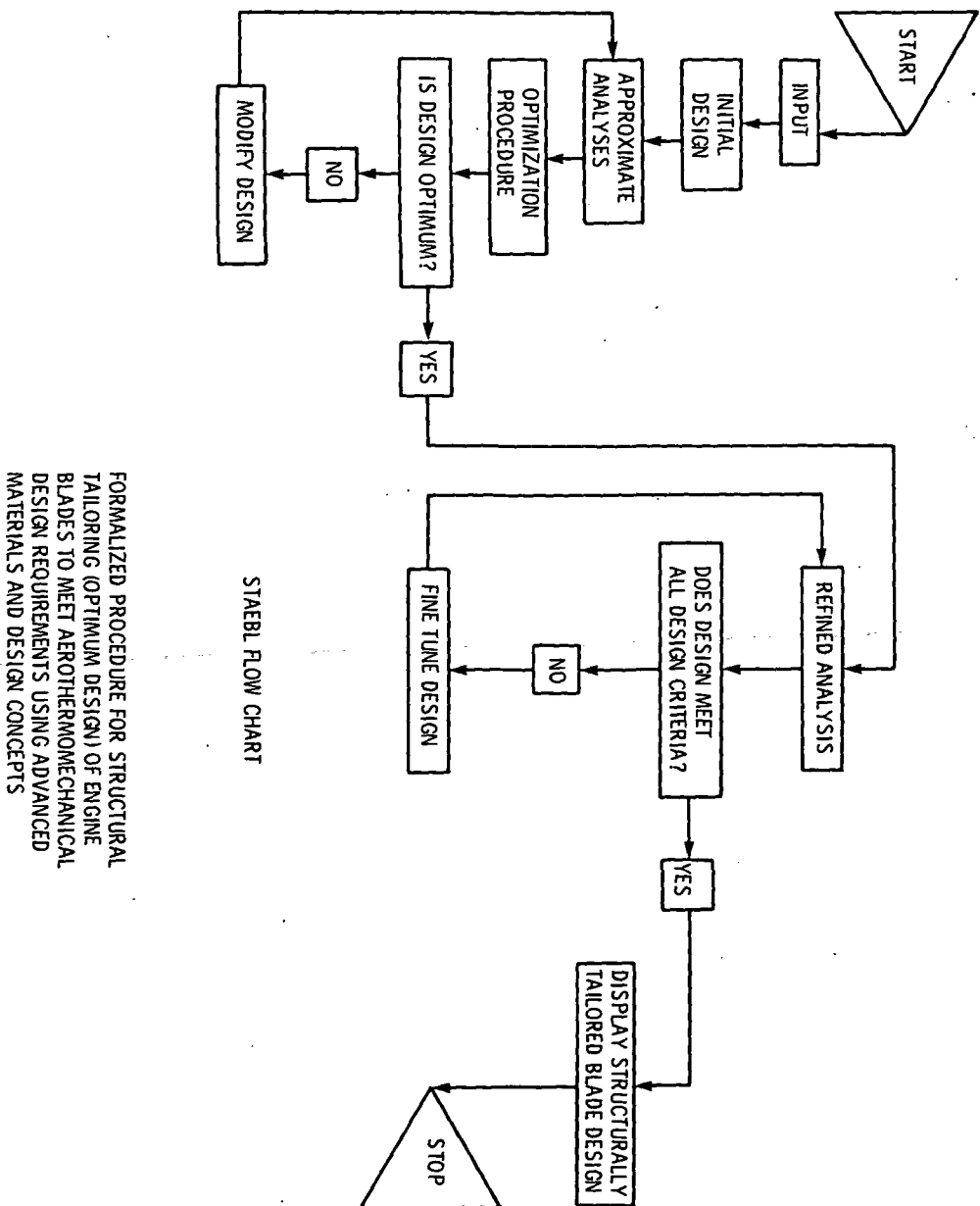
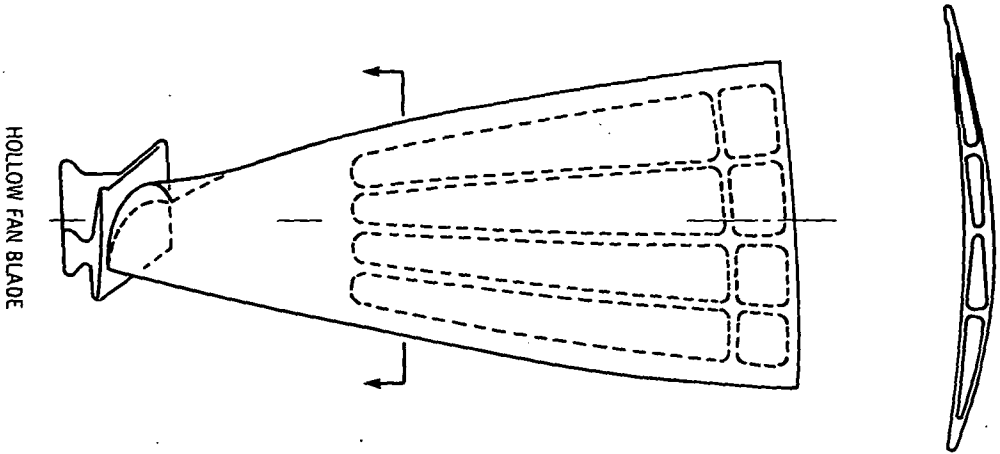


Figure 15. - Structural tailoring of engine blades (STAEBL).

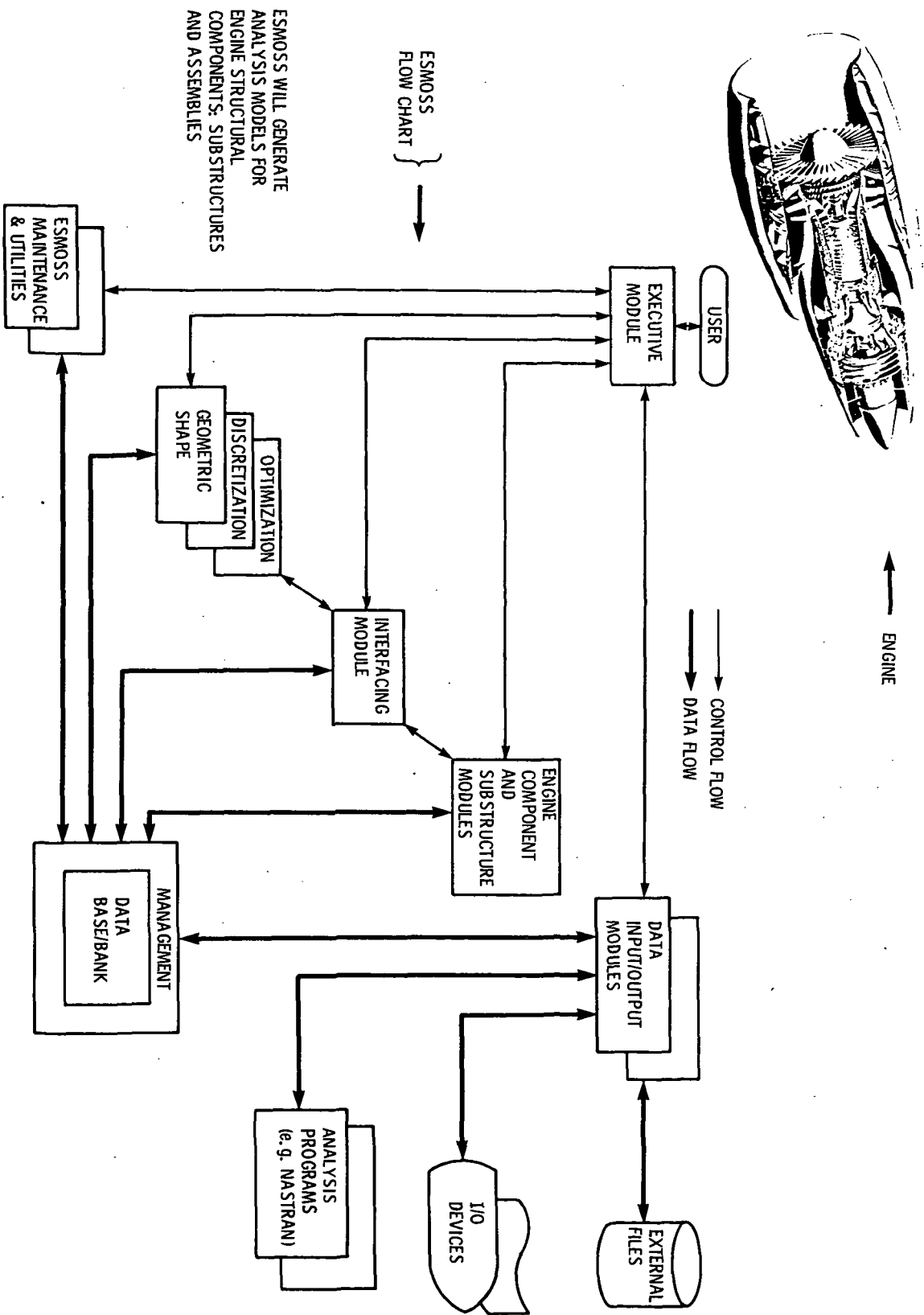


Figure 16. - Engine structures modeling software system (ESMOSS).

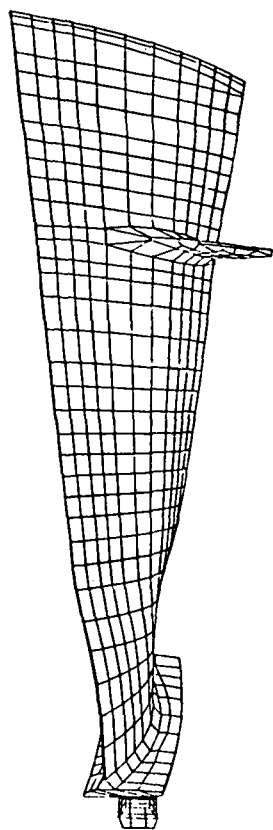


Figure 17. - Complete blade model.

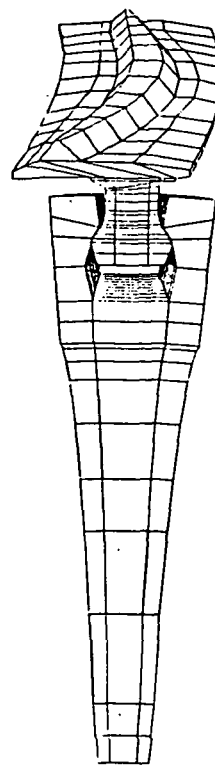


Figure 18. - Interfacing of blade and disk.

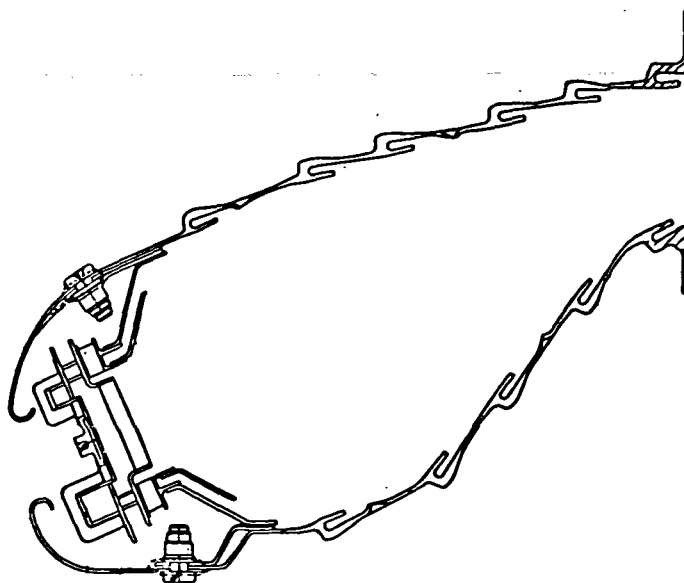


Figure 19. - Single wall combustor liner.

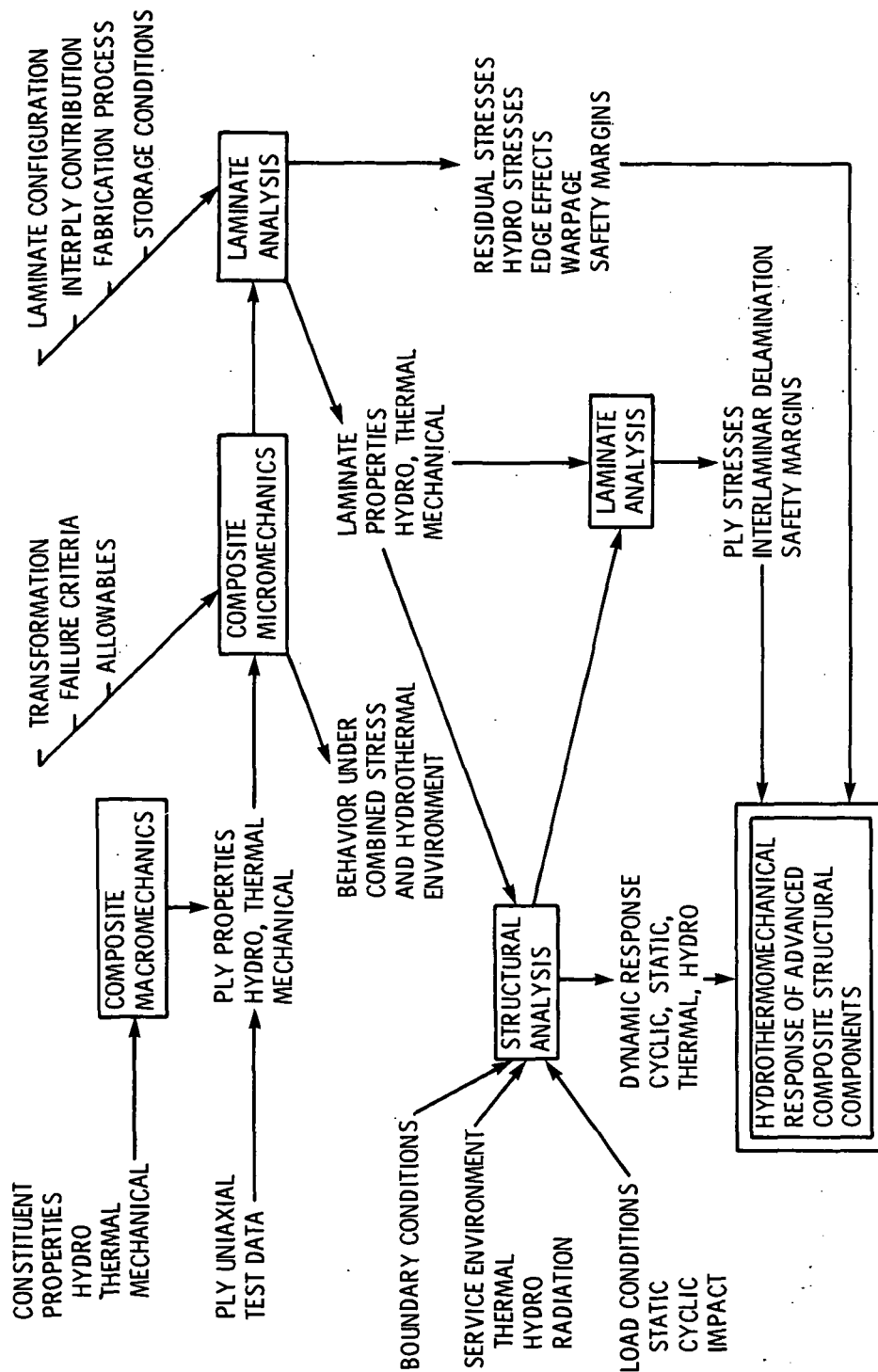


Figure 20. - Flow diagram of integrated theory for predicting the hydrothermomechanical response of advanced composite components.

1. Report No. NASA TM-82713	2. Government Accession No.	3. Recipient's Catalog No.	
4. Title and Subtitle INTEGRATED ANALYSIS OF ENGINE STRUCTURES		5. Report Date	
		6. Performing Organization Code 505-32-62	
7. Author(s) Christos C. Chamis		8. Performing Organization Report No. E-995	
		10. Work Unit No.	
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135		11. Contract or Grant No.	
		13. Type of Report and Period Covered Technical Memorandum	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D. C. 20546		14. Sponsoring Agency Code	
15. Supplementary Notes Prepared for the Winter Annual Meeting of the American Society of Mechanical Engineers, Washington, D. C., November 16-21, 1981.			
16. Abstract <p>The need for light, durable, fuel efficient, cost effective aircraft requires the development of engine structures which are flexible, made from advanced materials (including composites), resist higher temperatures, maintain tighter clearances and have lower maintenance costs. The formal quantification of any or several of these requires integrated computer programs (multilevel and/or interdisciplinary analysis programs interconnected) for engine structural analysis/design. Several integrated analysis computer programs are under development at Lewis Research Center. These programs include: (1) COBSTRAN-Composite Blade Structural Analysis, (2) CODSTRAN-Composite Durability Structural Analysis, (3) CISTRAN-Composite Impact Structural Analysis, (4) STAEBL-Structural Tailoring of Engine Blades, and (5) ESMOSS-Engine Structures Modeling Software System. Three other related programs, developed under Lewis sponsorship, are described.</p>			
17. Key Words (Suggested by Author(s)) Computer programs; Structural analysis; Design; Fiber composites; Impact; Durability; Optimization; Structural modeling; Integrated analysis		18. Distribution Statement Unclassified - unlimited STAR Category 39	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages	22. Price*

National Aeronautics and
Space Administration

Washington, D.C.
20546

Official Business

Penalty for Private Use, \$300

SPECIAL FOURTH CLASS MAIL
BOOK

Postage and Fees Paid
National Aeronautics and
Space Administration
NASA-451



NASA

POSTMASTER: If Undeliverable (Section 158
Postal Manual) Do Not Return
